

CFD test case 2

Turbulent water transport in a hydraulically smooth pipeline

Suggestions for setting up the case

Ing. Luca Nicola Quaroni

Ing. Jorge Soto

Ing. Federico Lanteri

Prof. Gianandrea Vittorio Messa

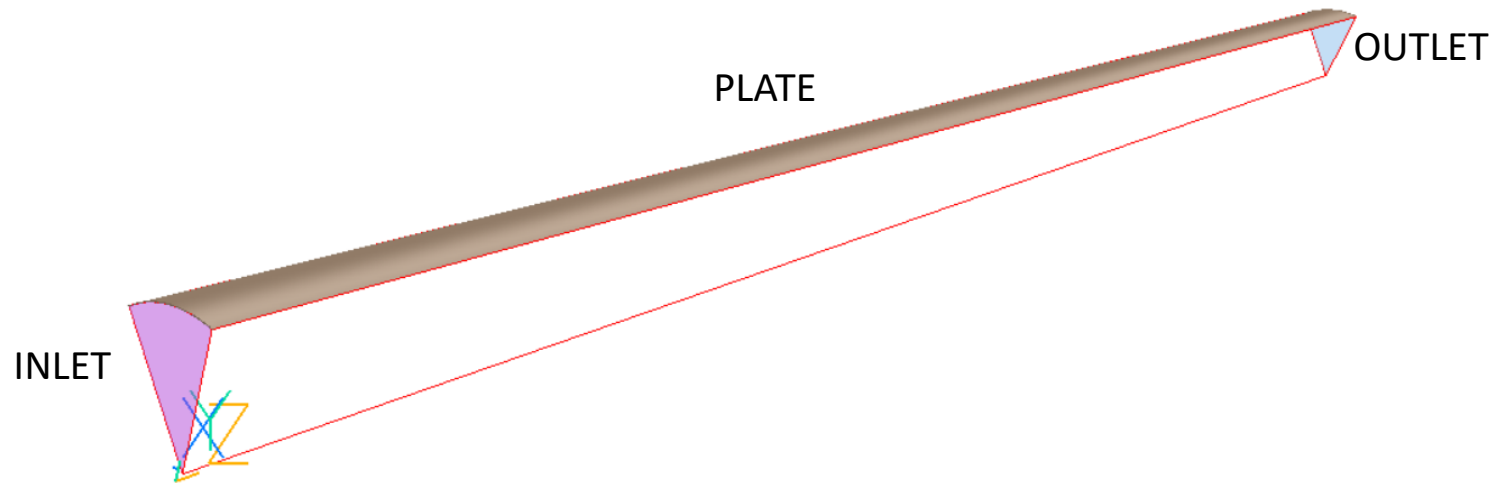


POLITECNICO
MILANO 1863

Numerical domain

Basic quantities

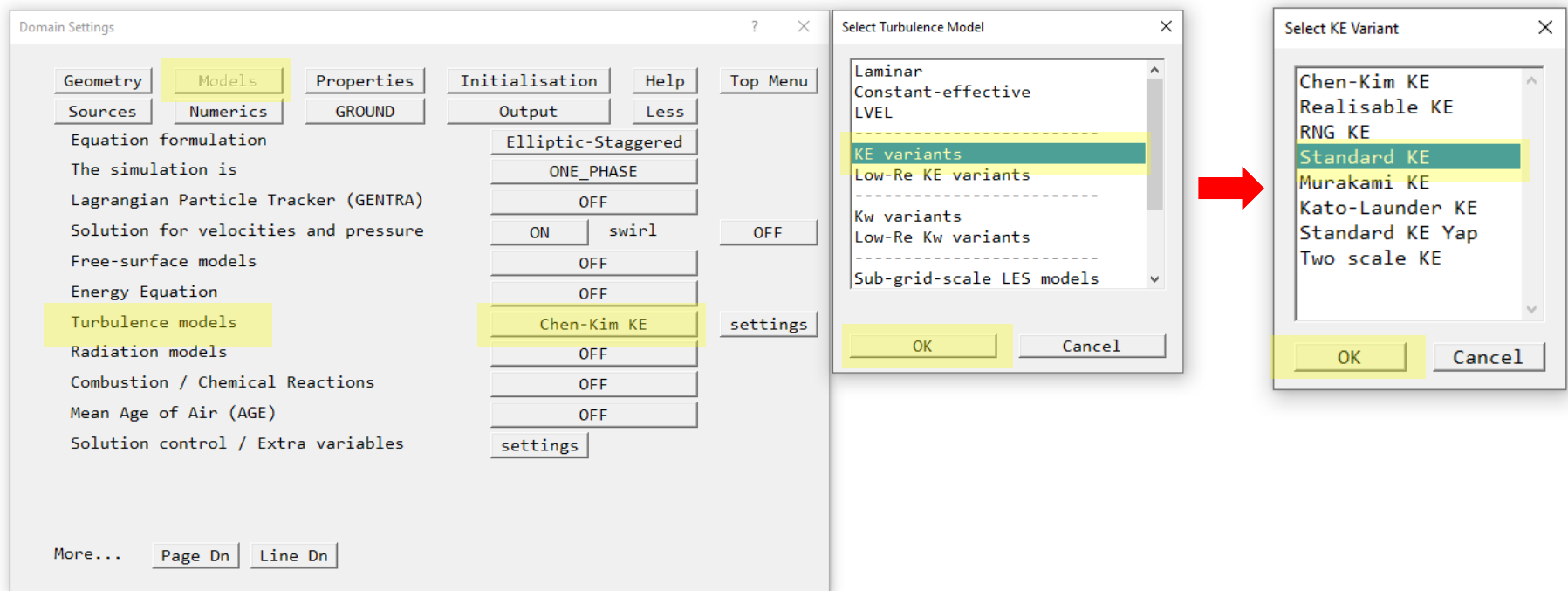
To set the numerical domain, the mesh and the fluid properties follow the same steps as in *Test Case 1*.



Numerical domain

Turbulence model

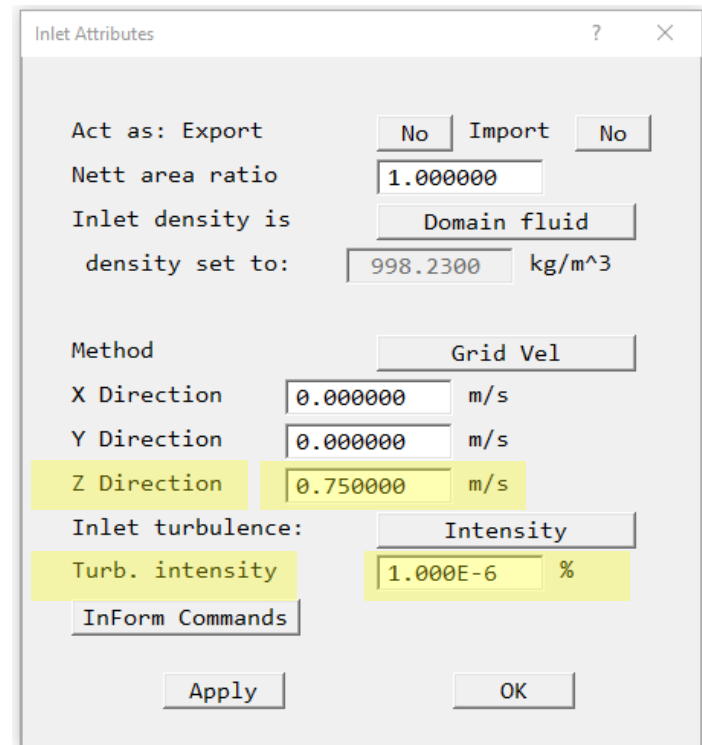
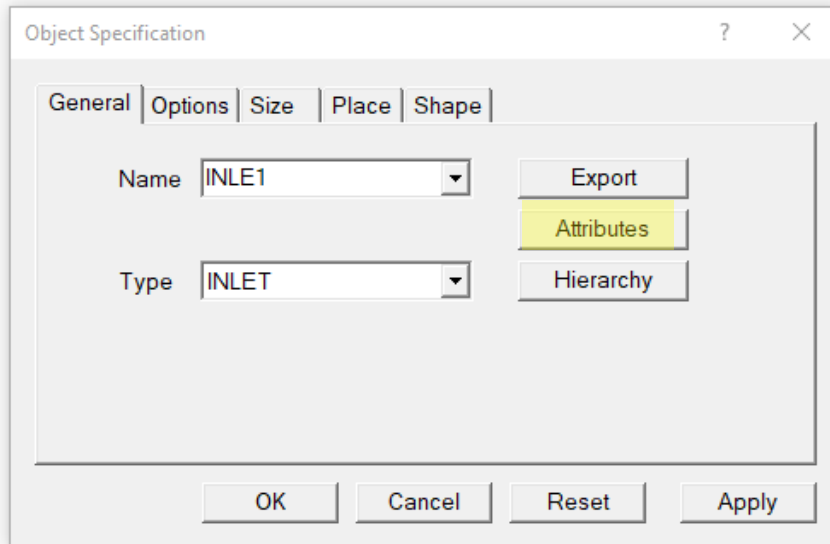
It is necessary to specify the type of flow as turbulent, modelled through the standard $k - \varepsilon$ model. Go to *Settings* → *Domain Attributes* → *Models* → *Turbulence Models*. Click on the current turbulence model → Select KE variants and click OK → Select the *Standard KE* model from the list and click OK.



Numerical domain

Boundary conditions: inlet

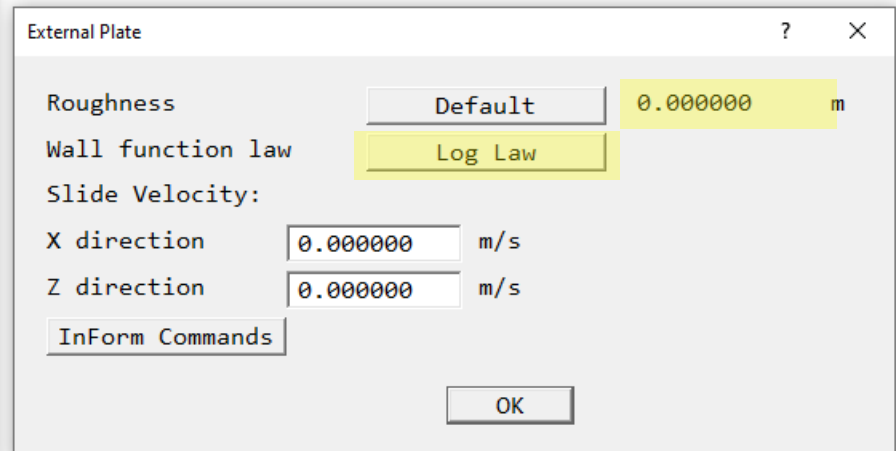
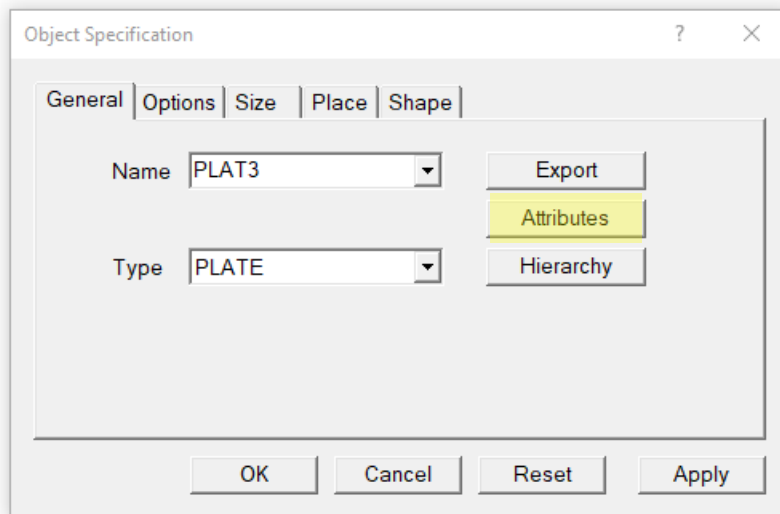
Go to *Settings* → *Object Attributes*. Create the *Inlet* and impose a Z-velocity equal to 0.75 m/s and an inlet turbulent intensity (TI) equal to 0.000001% (almost zero).



Numerical domain

Boundary conditions: plate

Go to *Settings* → *Object Attributes*. Create the *Plate*. Verify that the wall roughness is zero and the proper wall function law is selected.



Output

Derived variables

If YPLS (y^+) and STRS (τ_w/ρ) are not being stored, go to *Settings* → *Domain Attributes* → *Output* → *Derived variables*. Check that YPLS and STRS are set to ON.

Domain Settings

Derived Variables Printout Settings

Previous panel

Print wall coefficients

Print Y+ values

3D storage: Wall functions

Skin friction (SKIN) Stanton Number (STAN)

Shear stress (STRS) Yplus (YPLS)

Heat transfer coef (HTCO)

3D storage: Friction force components

X (SHRX) Y (SHRY) Z (SHRZ)

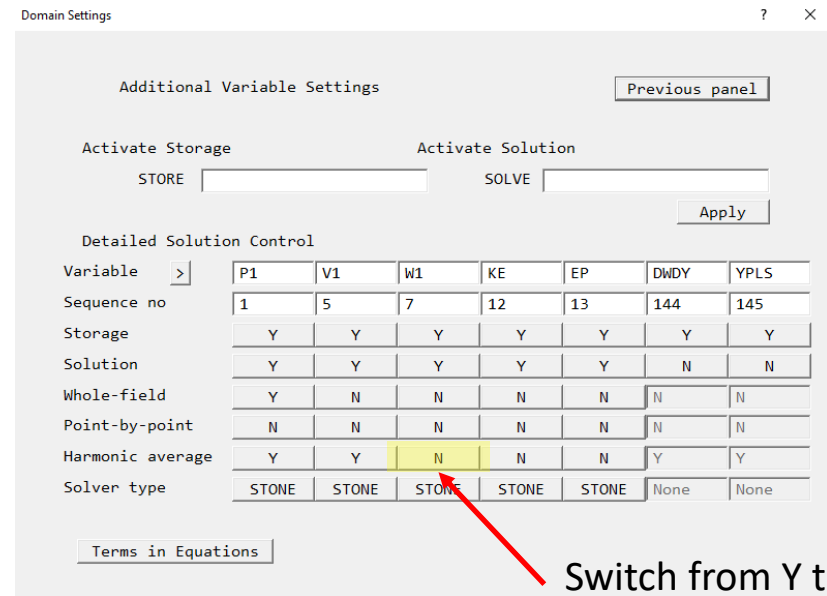
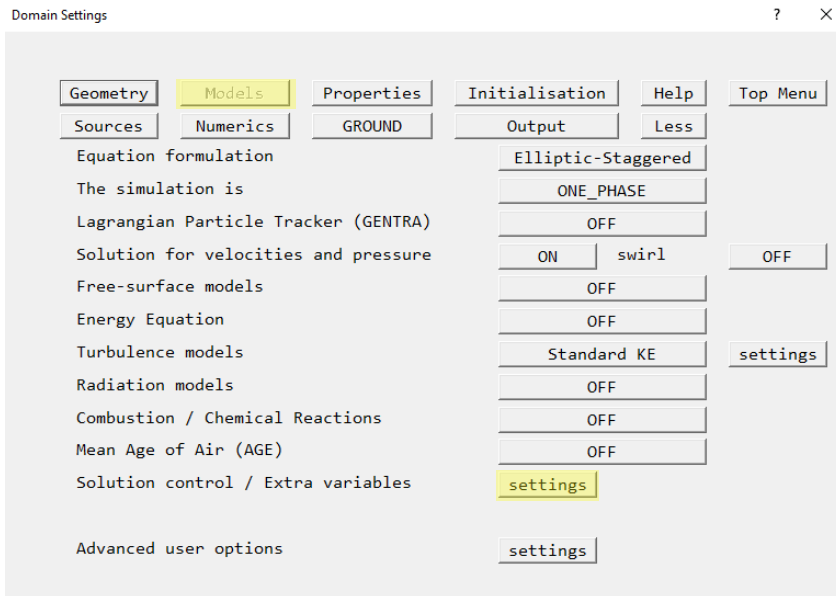
3D storage: Derived variables

More...

Output

Derived variables

The storage of variable DWDY can be activated as explain in the previous test case. The evaluation of the derivative in the first and last cells along the Y direction is quite critical for turbulent flow. In order to partially improve the accuracy, it is recommended to use arithmetic averaging for the diffusion coefficients of the variable W1.



Switch from Y to N
for variable W1

Post-processing

MATLAB import

To answer the questions provided in the *Requests*, it is advised to use MATLAB. Particularly, reference should be made to the script for importing the formatted files from PHOENICS (*XYZ_reduced_19_20.mat*) made available in the WeBeep page of the course.

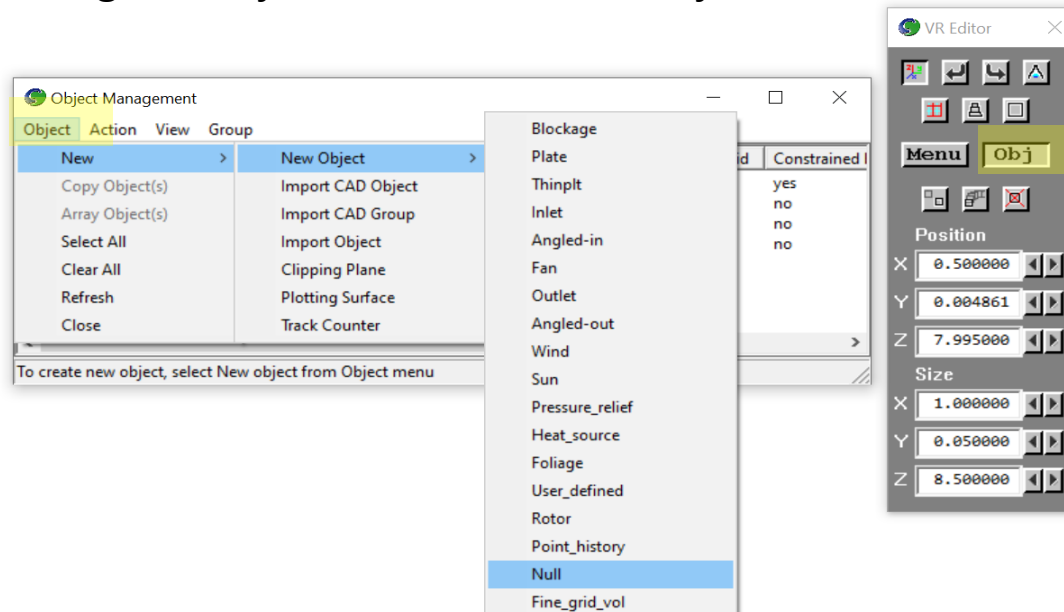
Import into MATLAB all variables you might need, e.g. KE, EP, ENUT, DWDY, STRS, YPLS, ...

```
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
%
%   IMPORT OF .PHI FILE IN MATLAB WORKSPACE
%
%
% VARIABLES SAVED IN WORKSPACE
% NX, NY, NZ           number of cells in the three directions
% X_E, Y_N, Z_H       coordinates of cell's faces
% X_C, Y_C, Z_C       coordinates of cell's centers
% xn3D, yn3D, zn3D    position of the cell's nodes in case of BFC simulation (N
% xc3D, yc3D, zc3D    position of the cell's nodes in case of BFC simulation (N
% P1, U1, V1, W1 ...  matrix (NXxNYxNZ) of the variables
% U1_C, V1_C, W1_C    matrix (NXxNYxNZ) of velocities interpolated in cell's cen
% XX_C, YY_C, XX_E, YY_N meshgrid
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
```


Numerical domain

Null object (*optional*)

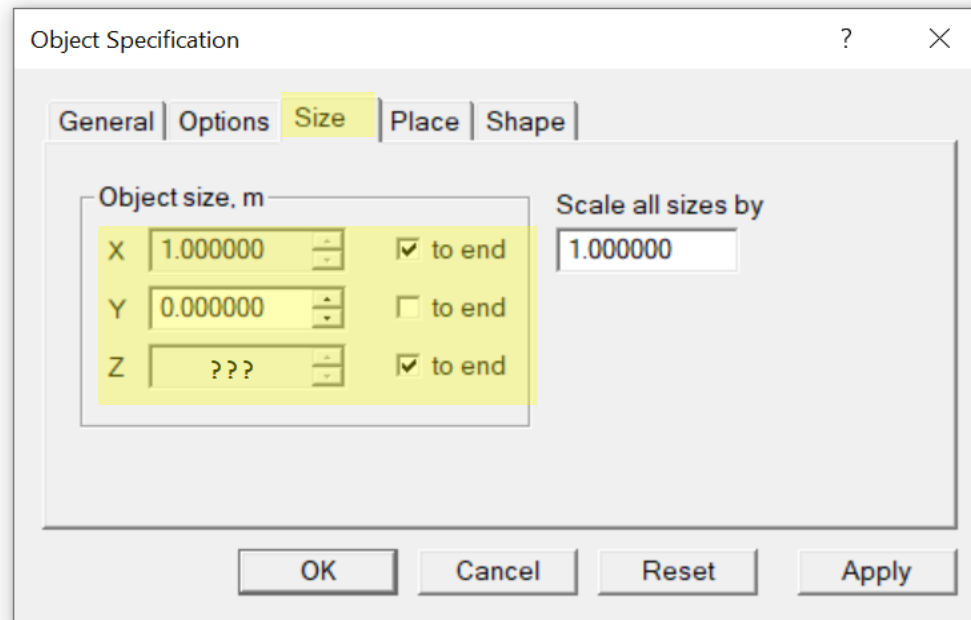
In PHOENICS, it is possible to adjust the size of the nearest cell by creating a *Null* object. *Null* objects divide the computational domain in two regions. For each region, it is possible to set different number of cells. The objective is to create a region close to the wall with only one cell of fixed size. Thus, when performing the grid independence study, the size of the nearest cell to the wall (size of y^+) remains invariable while the rest of the domain (number of cells) is modified. Click on *Obj* in VR Editor window. On the *Object Management* window, go to *Object* → *New* → *New Object* → *Null*.



Numerical domain

Null object (*optional*)

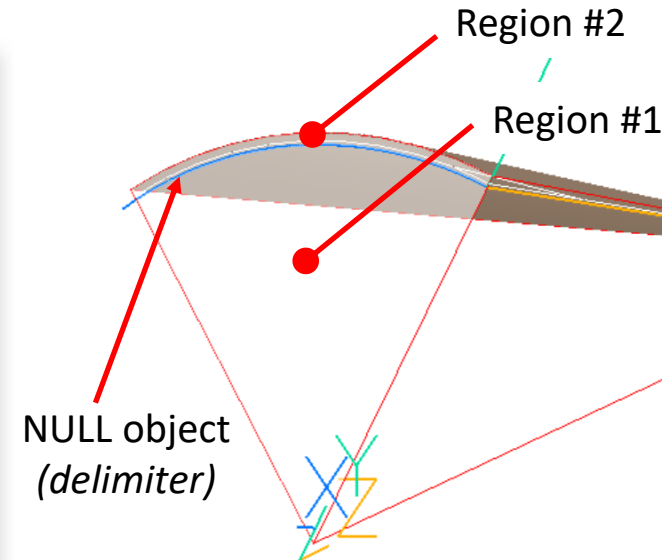
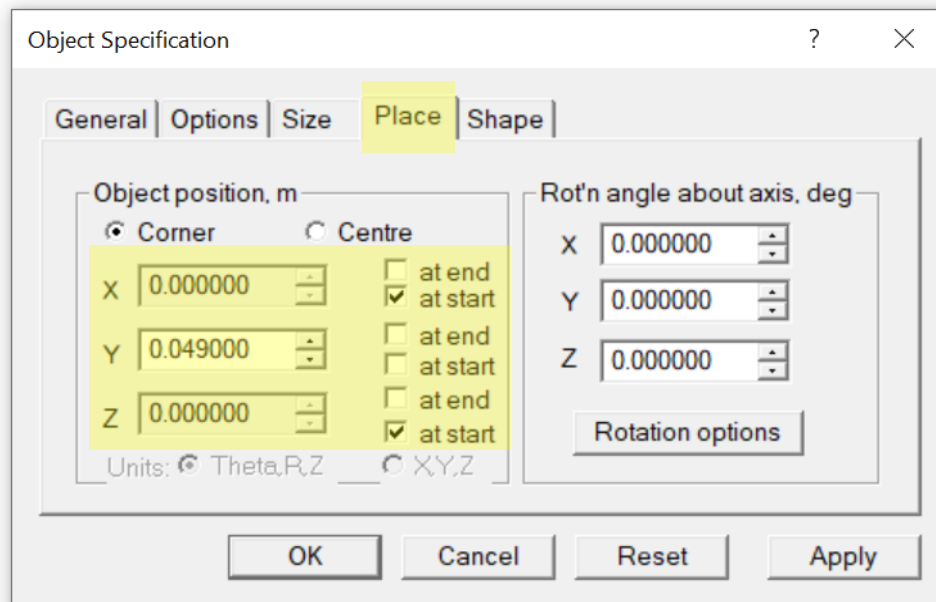
On the *Object Specification* window, go to *Size*. Set *Null* size to *end* for both *X* and *Z* directions, and zero for *Y*. This creates a plane in *Y* that separates the computational domain in two regions.



Numerical domain

Null object (*optional*)

On the same window, go to *Place*. Set *Null* position *at start* for both *X* and *Z* directions. In the *Y* direction, set the distance from the center where the domain will be divided. The size of the region close to the wall is equal to difference between the pipe *Radius* and the place where the *Null* object is emplaced in *Y*.



Numerical domain

Null object (*optional*)

Go to Setting → Domain Attributes → Geometry. Now, on *Grish Mesh Setting* window it is indicated that there are **two** regions in the Y direction. It is important to check if the region closest to the wall has only one cell. Go to *Edit all regions in Y* direction.

Number of regions
in you domain: 2

Edit all regions in Y
direction

Co-ordinate system		Time dependence		
Cylindrical-polar		Steady		
Inner radius	0.000000 m			
Cut-cell method	SPARSOL	Settings		
	X-Manual	Y-Manual	Z-Manual	
Domain size	1.000000	0.050000	???	m
Domain origin	0.000000	0.000000	0.000000	m
Number of cells	1	11	100	
Tolerance	1.000E-3	1.000E-5	1.000E-5	m
No of regions	1	2	1	
Modify region	1	1	1	
Size	1.000000	0.049000	???	
Distribution	Power law	Power law	Power law	
Cell power	Free	Set	Set	
Cells in region	1	10	100	
Power/ratio	1.000000	1.000000	1.000000	
Symmetric	No	No	No	
Edit all regions in	X direction	Y direction	Z direction	
Total number of cells is 1100				
Cancel		Apply	OK	

Numerical domain

Null object (*optional*)

On *Y direction settings* window it is possible to edit the number of cell of each region. Adjust the number of cell in region #1 to perform the grid independence analysis. Set the number of cells in region #2 equals to one. Click Apply and then Ok. On *Grid Mesh Setting* windows, click Apply and then OK.

Y direction settings

Global settings:-

Domain size m Number of cells

Auto grid settings:-

Min cell fraction % Boundary - Low High

Region settings:- (Currently 2 regions)

Free all regions Tolerance m

Reg	End positn	Cells	Distributn	Power	Symmetric	Cell powr
1	<input type="text" value="0.049000"/>	<input type="text" value="10"/>	<input type="button" value="Power law"/>	<input type="text" value="1.000000"/>	<input type="button" value="No"/>	<input type="button" value="Set"/>
2	<input type="text" value="0.050000"/>	<input type="text" value="1"/>	<input type="button" value="Power law"/>	<input type="text" value="1.000000"/>	<input type="button" value="No"/>	<input type="button" value="Set"/>

Reg #1: Number of cells
in the rest of the domain

Reg #2: Number of cells in
the region closest to the wall

Numerical domain

Null object (*optional*)

Remember:

- i. The size of region #2 is modified by adjusting the place of the *Null* object.
- ii. After running a simulation, verify if the size of region #2 fullfil the conditions of y^+ (See next slide to store YPLS array):
 - For wall function turbulence models: $30 \leq y^+ \leq 130$
 - For low Reynolds turbulence models: $y^+ \approx 1$.
- iii. For the grid-indepencece study, modify the number of cells in region #1.